

CFD Study of Exhaust Gas Energy Recovery Potential of a Radial Flow Turbine for Electricity Generation

G.N. Nwaji^{1*}, I. Onyenezido¹, C. Ononogbo², P.O. Anadebe³, K.T. Ajayi⁴,
E.E. Anyanwu¹

¹Federal University of Technology, Owerri, Imo State, Nigeria.

²University of Agriculture and Environmental Sciences, Umuagwo, Imo State, Nigeria.

³University of Nigeria, Nsukka, Enugu State, Nigeria.

⁴University of Lagos, Akoka, Yaba, Lagos, Nigeria.

ABSTRACT: The increasing demands for low-fuel, highly efficient vehicles with low CO₂ emissions have forced the auto-manufacturing industry and other researchers to continuously seek for innovative ways of improving engine performance and passenger comfort without sacrificing safety and increasing operational costs of vehicles. This study, which represents such innovative efforts, employed computational fluid dynamics (CFD) to analyze the flow variables of fluids flowing through a radial turbine, and the dynamic characteristics of the fluid within the turbine. The results obtained indicate that a radial turbine can be used to harvest and convert the waste exhaust gas energy from the internal combustion (IC) engine into useful work, as about 70% of exhaust gas energy was extracted with an isentropic efficiency of 84%. Under ideal operational situations, 59.77 kW of electric power can be produced by the electric generator coupled to the turbine. The study results also show that it is technically possible to provide a separate power source for engine accessories such as water and oil pumps, air conditioner compressors, cooling fans, and alternators in an automobile by using a radial turbine to harness the exhaust gas energy and convert it into useful work for the generation of electricity. This can be achieved by installing a radial flow turbine in the exhaust manifold of an IC engine of an automobile and coupling an electricity generator to the radial turbine shaft. This will serve as a good alternative to the use of expensive superchargers for extra energy requirements in automobiles for example in order to satisfy the energy demands for the automobile's accessories, for improved passenger comfort, with the possible reduction in the cost of fuel and impact of gas emissions to the environment together with a significant improvement in the automobile engine service life.

KEYWORDS: Exhaust gas, Energy recovery, Radial flow turbine, Electrical energy, IC engine, Isentropic efficiency, Fluid dynamics, Electricity generator.

<https://doi.org/10.29294/IJASE.9.2.2022.2758-2772>

©2022 Mahendrapublications.com, All rights reserved

1. INTRODUCTION

The increasing demand for energy due to the ever-increasing world population and the increasing agitation for a safer environment have given rise to the need for increased sustainable energy generation [1]. Furthermore, the high cost of fuels required to generate power to operate production equipment and machines is also driving research on internal combustion (IC) engine efficiency optimization. Improved conversion of energy in engines gives rise to reduced energy costs globally [2, 3]. Lee et al., [4] reported that the waste heat recovery has proved viable in improving the efficiency of IC engines. It has become accepted as an effective means of conserving finite fossil fuel

reserves as well as helping to minimize environmental pollution. The first is achieved by minimizing energy waste (which translates to reduced fuel consumption); while the other is obtained through a carbon footprint reduction strategy. Based on the emerging research trends on the recovery of heat from exhaust gas for efficiency improvement in IC engines, the depletion of fossil fuel reserves which is associated with high global energy demands would be minimized, leading to a considerable reduction in environmental damage [5-9]. Reports in the literature indicate that about 70 % of the total energy produced from IC engine in-cylinder combustion goes to the environment

*Corresponding Author: godswillmee@gmail.com

Received: 18.09.2022

Accepted: 17.10.2022

Published on: 01.11.2022

Nwaji et al.,

as heat [10,3]. The heat balance of an IC engine splits the energy output into approximately three equal parts, viz, the energy converted into useful work, the part lost to the surroundings through the exhaust gas, and the part lost to the engine coolant. Pradip et al., [11] reported that thermal losses associated with the total energy produced in an IC engine through exhaust gases are in the range of 25–30%. These heat losses through the exhaust gas could go up to 40% [12,13], which is a huge waste. However, to ensure a substantial reduction of fossil fuel consumption in order to minimize its negative impact on the environment, manufacturers now place priority on either renewable energy facilities or try to reduce energy waste in IC engines, using waste heat recovery (WHR) techniques [14,15]. Reshid et al., [16] observed that a significant amount of heat can be recovered from the exhaust gas of an IC engine to take care of other thermal needs, thus improving energy efficiency in the range of 30 - 50%.

The three important components of WHR technology are a source of the heat (which must be accessible), recovery technique and application for the recovered waste heat [17]. The technique of heat recovery depends on the waste heat temperature and the economics involved. There are numerous Technologies available for WHR. However, there is a wide potential for their applications, which are yet to be realized in industries [18]. WHR methods include heat transfer between fluids, mechanical or electric power production, utilization of waste heat for the production of cooling or heating effects, etc. [19-21].

Many technologies such as heat exchangers, thermoelectric devices, turbines, etc., have been employed effectively for WHR, thus increasing the energy utilization efficiency of IC engines, while reducing the emission of pollutants. A significant portion of engine power is utilized by the accessories like air conditioners, alternators, water and oil pumps, etc. These accessories can be coupled to the waste energy recovery system instead of a straight connection to the engine, thereby improving the power available to the wheel, and reducing the consumption of fuel. One of the promising techniques for achieving that is the use of a radial turbine. A radial turbine is a turbine in which the flow of the working fluid is radial to the shaft axis. It has been reported that radial

turbines can work with relatively higher-pressure ratios per stage with lower flow rates. The stage output work in a single-disk radial gas turbine that operates on a direct radial flow scheme without axial turning is less than the isentropic enthalpy drop due to the aerodynamic losses in the turbine stage. For many years, different vehicle manufacturers have engaged in the design of efficient radial or centrifugal turbines for different purposes in vehicles. Currently, radial flow turbines (RFTs) are used mostly in automobile engines, for turbocharger applications [22]. This has resulted in a significant improvement in the overall efficiency of automobile engines. When a radial flow turbine is installed in the exhaust manifold of an automobile IC engine, the actual transfer of energy between the exhaust gas and the turbine takes place in the space in between the impeller inlet and outlet (i.e., the volute) of the radial turbine. The IC engine by the nature of its operation generates continuously pulsating exhaust gas flow and it is reported in [23], that turbine behaviour under the highly unsteady flow deviates from its equivalent on steady Performance. The volute section is a critical part of a radial turbine and oftentimes, the geometry of the volute and other elements within it, determines how efficient, the fluid flow and the energy conversion process in the turbine will be. In a radial inflow turbine, the volute has to be carefully designed to ensure that the desired rotor inlet conditions for optimal performance, like the absolute Mach number, flow angle, mass flow parameter (MFP), efficiency and so on, are achieved [24]. The volute determines how fluid is fed into the turbine impeller or rotor. Yang et al., [23] reported the primary function of a volute in radial, as well as mixed-flow turbine, is to feed the rotor with the uniform flow at the desired flow angle, in order to optimize the turbine performance. Several other authors have also reported on the impact of the volute geometry on the efficiency and other fluid flow characteristics within a turbine. Shah et al., [24], designed and conducted a numerical simulation of radial inflow turbine volute with three different cross-sectionally shaped volutes, using a commercial computational fluid dynamics code, and concluded that based on the maximum pressure-loss coefficient, that the circular cross-section gave the best efficiency compared to the other types of volute cross-sections.

To determine the exhaust waste energy recovery potential of a radial turbine coupled to the exhaust manifold of an automotive engine requires extensive research targeted at assessing the viability or otherwise, of the proposed system. This no doubt will require enormous resources in the area of the development of the device and its setup for experimental investigation. However, in order to cushion the effect of cost, time and other technical constraints associated with real life experimental studies, numerical analysis, which has proven to be a viable and cost-effective alternative is now being deployed for such essential preliminary investigations[25-27]. Furthermore, Yang et al., [23] reported both experimental and numerical investigations on the effect of volute cross-sectional shape on radial turbocharger turbine under pulsating conditions in internal combustion engine using two different volutes. The authors observed that the turbine with the optimized volute gave a better cycle averaged efficiency under pulsating flow conditions for different loadings and frequencies, and concluded that the simplified CFD method can reasonably predict the performance of turbine under pulsating flow conditions. The present research is aimed at conducting a computational fluid dynamic (CFD) modelling and numerical simulation of the exhaust gas energy recovery potential of a radial flow turbine for electricity generation. Furthermore, an analysis of the dynamic characteristics of the exhaust gas flowing through the volute and the profile of the radial turbine wheel were conducted, with substantial effort devoted to determining the energy extraction efficiency of the turbine.

2.0 METHODOLOGY

The continuity, momentum and energy (CME) equations governing fluid motions are solved in 3D turbine geometry generated using Autodesk Inventor. The $k - \omega$ two equation model is used to capture the turbulence in the system and close the problem, with the equation of state for ideal gas. The model formulation is based on the following assumptions (i) the flow is a single fluid flow considered as a continuum (ii) there is no volumetric energy generation within the control volume (iii) the thermo-physical properties are gas temperature-dependent throughout the flow, and (iv) there is no combustion occurring in the turbine stage.

2.1 Governing Equations of Fluid Dynamics

2.1.1 Continuity Equation

Considering an infinitesimal fluid element, the conservative form of the continuity equation is given by (Eqn 1):

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_j)}{\partial x_j} = 0 \quad \text{---(1)}$$

The Reynolds Averaged Navier-Stokes (RANS) approach has become prominent in CFD studies of complex flows, as it is able to resolve to a greater degree the turbulent length scales, when closed with an appropriate turbulence model.

2.1.2 Momentum Equation

Using the Reynolds-averaged approach, the momentum equation may be written in a condensed form as:

$$\rho \left(\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_k \frac{\partial \bar{u}_i}{\partial x_k} \right) = - \frac{\partial \bar{P}}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu \frac{\partial \bar{u}_i}{\partial x_j} \right) + \frac{\partial R_{ij}}{\partial x_j} \quad (2)$$

where R_{ij} is the Reynolds stress tensor given as:

$$R_{ij} = -\rho \overline{u'_i u'_j} \quad \text{----(3)}$$

The averaging process normally introduces another stress known as the Reynolds stress and must be taken care of to further close the problem for the convergence of the solution. The Boussinesq Model or Approximation is employed in resolving the stress tensor, and is given as:

$$R_{ij} = -\rho \overline{u'_i u'_j} = \mu_T \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \mu_T \frac{\partial \bar{u}_k}{\partial x_k} \delta_{ij} - \frac{2}{3} \rho k \delta_{ij} \quad \text{----(4)}$$

where μ_T is the eddy or turbulent viscosity and δ_{ij} is the Kronecker delta.

2.1.3 Energy Equation

The applied energy equation is given as:

$$\frac{\partial(\rho T)}{\partial t} = \frac{\partial}{\partial x_j} \left(\frac{\lambda}{c_p} \left(\frac{\partial T}{\partial x_j} \right) \right) - \frac{\partial(\rho u_j T)}{\partial x_j} + \frac{\rho T}{c_p} \left(\frac{\partial v}{\partial T} \right)_p \frac{DP}{Dt} - \frac{\tau_{ij}}{c_p} \frac{\partial u_j}{\partial x_j} \quad \text{----(5)}$$

2.1.4 Turbulence modelling

Turbulence is the unsteady and irregular movement of transported bodies in space within a defined time. Modelling of turbulence will ensure resolution of the turbulent viscosity, μ_T . For turbo-machinery analysis, the widely used model is the standard $k - \omega$ model. This is because, the model resolves wall effects without recourse to any damping function and has high

accuracy and robustness in handling boundary flows with pressure gradients.

The standard $k - \omega$ model resolves the transport equation for the kinetic energy (k) and the dissipation rate (ω) in a manner that:

$$\mu_T = f \left(\frac{\rho k}{\omega} \right) \quad \text{---(6)}$$

$$\mu_T = \alpha^* \frac{\rho k}{\omega} \quad \text{---(7)}$$

The k and ω equations are given, respectively as Equations 8 and 9[28]:

$$\rho \frac{Dk}{Dt} = \tau_{ij} \frac{\partial \bar{u}_i}{\partial x_j} - \rho \beta^* f_\beta k \omega + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \quad \text{---(8)}$$

$$\rho \frac{D\omega}{Dt} = \alpha^* \frac{\omega}{k} \tau_{ij} \frac{\partial \bar{u}_i}{\partial x_j} - \rho \beta f_\beta \omega^2 + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial \omega}{\partial x_j} \right] \quad \text{---(9)}$$

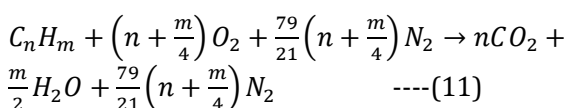
To complete the set of equations and account for all the unknowns, the equation of state for ideal gases was employed. The ideal gas equation of state (Eqn. 10) is sufficient for this modelling surface, since it gives a good approximation of the behaviour of real gases at conditions below the critical conditions:

$$P = \rho RT \quad \text{---(10)}$$

where P is the pressure, ρ is the density of gas, R is the specific gas constant, and T is the temperature of the working fluid.

2.1.5 Working fluid model

The working fluid is the exhaust gas from a GT-Power turbocharged gasoline engine, running at 3500rpm. An engine working at full load was used, hence a rich mixture was considered, and for which reason there was no excess oxygen in the exhaust gas. Hence, the applicable reaction chemistry is shown in (Eqn. 11):



In order to properly define the thermodynamic properties of the exhaust gas, the following correlations were employed:

$$\rho = 353 T_g^{-1} \quad \text{---(12)}$$

$$c_p = 962.097 + 0.1507 T_g \quad \text{---(13)}$$

$$K_{CN} = 8.459 \times 10^{-3} + 5.7 \times 10^{-5} T_g \quad \text{---(14)}$$

$$\mu = 1.384 \times 10^{-5} + 2.68 \times 10^{-8} T_g \quad \text{---(15)}$$

where ρ is the density; c_p , the heat capacity; K_{CN} , the thermal conductivity; μ , the dynamic viscosity, and T_g , the temperature, of the exhaust gas.

2.2 Performance calculations

The thermodynamic and flow properties of the turbine were at specific points in the flow geometry using the function calculator imbedded in the solver, and the results are shown in table 1. With the knowledge of total enthalpy at the inlet and outlet, as shown in table 1, the turbine isentropic efficiency, shaft power, generator frequency, voltage, and electrical power were computed.

The actual power produced by the turbine is given as:

$$P_T = \eta_T \cdot \dot{m}_T (h_{o_1} - h_{o_2}) \quad \text{---(16)}$$

Where P_T is the Actual turbine power, η_T is the isentropic efficiency of the turbine, $\dot{m}_T$ is the mass flow rate into the turbine and $h_{o_1} - h_{o_2}$ is the total enthalpy drop across the turbine, which is given in table 1.

From the relation:

$$P_T = T \omega \quad \text{---(17)}$$

Where T is the torque on all 12 blades of the turbine (4.06N) and ω is the angular velocity in rad/s, given as:

$$\omega = \frac{2 \times \pi \times N}{60} \quad \text{---(18)}$$

Where N is the turbine rotational speed in rpm (140,539 rpm). Therefore,

$$\omega = \frac{2 \times \pi \times 140593}{60} = 14,722.86 \text{ rad/s};$$

And the actual power, $P_T = 4.06 \times 14,722.86 = 59,774.81 \text{ W}$.

The isentropic power, $P_{T_{is}}$ of the turbine can be expressed as:

$$P_{T_{is}} = \dot{m}_T (h_{o_1} - h_{o_2}) \quad \text{---(19)}$$

$$= 0.12 \times (863,342 - 269,172) = 71,300.4 \text{ W}$$

Therefore, isentropic efficiency is computed as:

$$\eta_T = \frac{\text{Actual Power of Turbine}}{\text{Isentropic Power of Turbine}} \quad \text{---(20)}$$

$$\eta_T = \frac{59,774.81}{71,300.4} = 0.84$$

From Electrical generator analysis, the frequency of electric generator f , is given as:

$$f = \frac{N \times P}{120} \text{ ----(21)}$$

Where N is the rotational speed of the shaft and P is the number of poles in the generator. It follows that:

$$f = \frac{140593 \times 4}{120} = 4686.43 \text{ Hz}$$

Also, the source voltage of the generator, V_{emf} is given as:

$$V_{emf} = k_i \times \omega \text{ ---(22)}$$

Where k_i is a torque constant given as 1.5 Nm. Hence, the source voltage becomes:

$$V_{emf} = 1.5 \times 14722.86 = 22,084.29 \text{ V}$$

Assuming there is no electrical and shaft losses, the actual shaft power will be equal to the electrical power developed. Therefore, the electrical power to be developed by a suitable generator will be 59.77 kW.

Table 1: Thermodynamic and flow variables at different regions in the flow geometry

VARIABLES	REGIONS			
	Turbine Inlet	Turbine Outlet	Diffuser Outlet	Outlet Pipe Outlet
Mass flow-rate (kg/s)	0.12	-0.12	-0.12	-0.12
Average fluid velocity(m/s)	76.59	447.47	308.404	21.64
Average static temperature(K)	1064.39	438.49	1021.02	1069.95
Average total temperature (K)	1067	537.86	1065.35	1070.16
Average static pressure (Pa)	138,447	19,157.7	101,640	101,325
Average total pressure (Pa)	139,419	56,093.9	118,105	101,403
Average static enthalpy(J/kg)	860,406	157,582	811,714	866,651
Average total enthalpy(J/kg)	863,342	269,172	861,457	866,885
Average static entropy($J/kg.K$)	1428.95	428.904	1382.1	1434.8

2.3 Geometry modelling, importation and mesh generation

A suitable geometry of a turbocharger turbine volute and blade was selected from Garrett Turbocharger catalogue and modelled using

Autodesk Inventor. This forms the flow domain in which the relevant equations are solved. The orthographic view of the turbine volute assembly is shown in Figure 1.

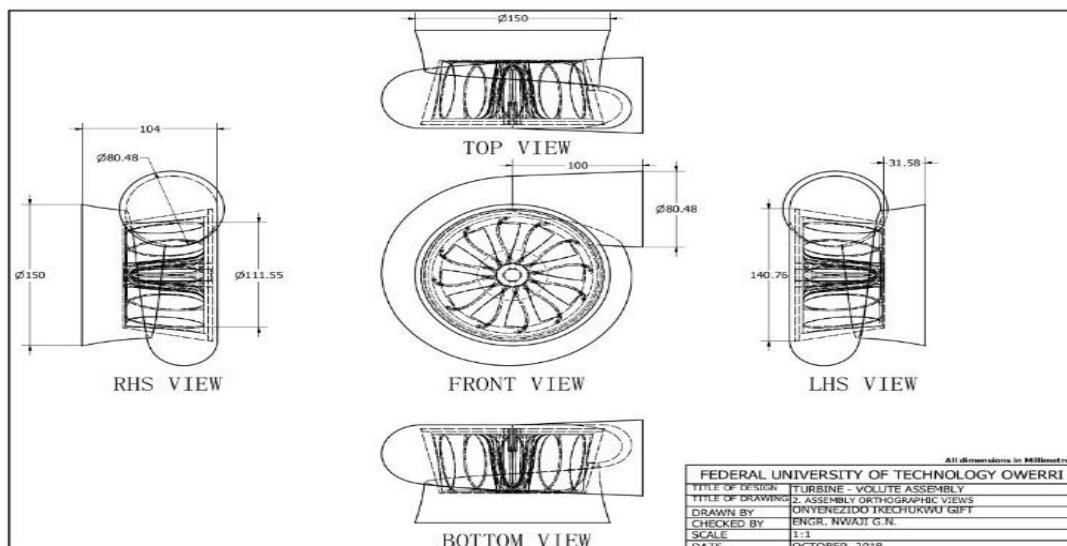


Fig. 1: Orthographic view of turbine-volute assembly

The geometry was converted into step file format and imported into CFX. Relevant parts were renamed, and the inlet and outlet name selections were created as shown in Fig. 2(a).

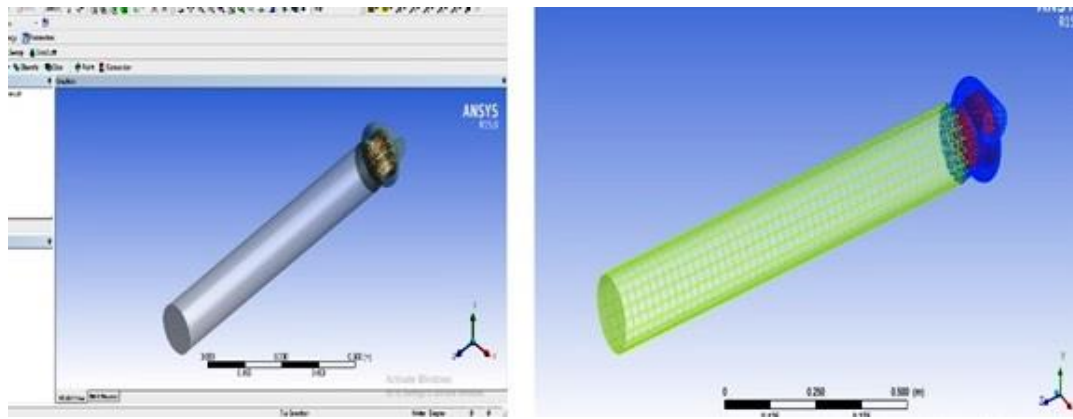
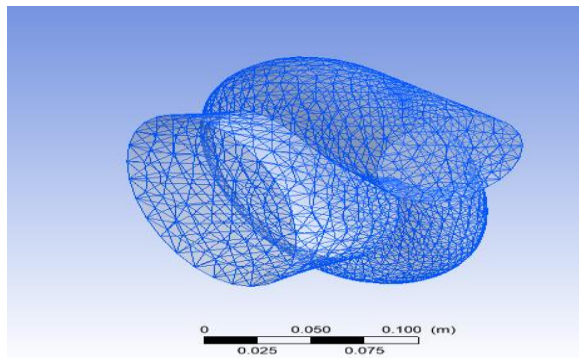


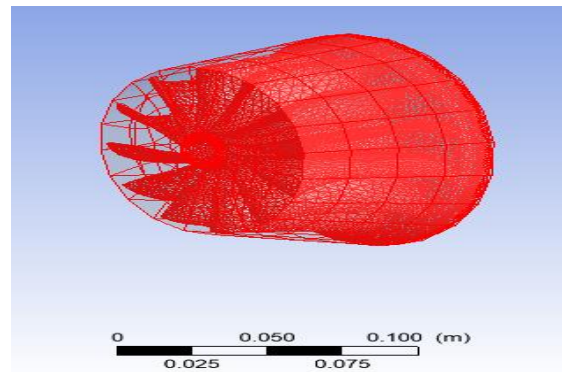
Fig. 2: (a) Geometry import into ANSYS (b) Mesh generation on turbine-volute assembly

The geometry was meshed automatically, generating a total of 146703 finite volume elements and 37096 nodes (see Fig. 2b and Table 2). Mesh cells for the volute and turbine blades are shown in Figures 3 (a) and (b).



(a)

Fig. 3: (a) Volute mesh cells



(b)

(a) turbine blade mesh cells

Table 2: Distribution of volume elements

Type		Distribution		
Element type	Number	Region	Elements	Nodes
Tetrahedral	144496	Volute	25165	4998
Wedges	8	Turbine	280	400
Pyramids	0	Turbine	119333	29202
Hexahedral	2199	Outlet Pipe	1927	2496
Polyhedral	0	Total	146703	37096
Total	146703			

2.4 Grid sensitivity

Grid sensitivity tests were carried out by observing the torque during the iteration at varying cell sizes. As the mesh was progressively made finer, the solution became unstable with minimal variations in the results. The solution became independent of the grid distribution at 146,703 elements and 37,096 nodes. Even though this grid size is coarse, the

solution remained stable throughout the iteration.

2.5 Simulation set-up and boundary conditions

The setup comprises three fluid domains, namely, the volute, turbine and Outlet pipe regions. The turbine blade stands as an immersed solid in the turbine region. Frame reference motion was defined as multiple

reference frame rotor. In this case, the mesh does not truly move, but the coordinate system rotates with the rotor, such that the flow equations are implemented in this reference frame (Galindo et al 2013a). Both the turbine

and turbine zone (i.e. region of fluid surrounding the turbine) is given a rotational speed of 140593rpm. The boundary conditions are summarized in Table 3.

Table 3: Applied boundary conditions

BOUNDARY CONDITION	PARAMETER	VALUE
Inlet	Total Temperature	1067K
Inlet	Mass flow rate	0.12kg/s
Outlet	Static Pressure (Atmospheric)	101325Pa
Turbine & Turbine zone rotation	Angular Velocity	140593rpm
All regions	Wall	No slip wall

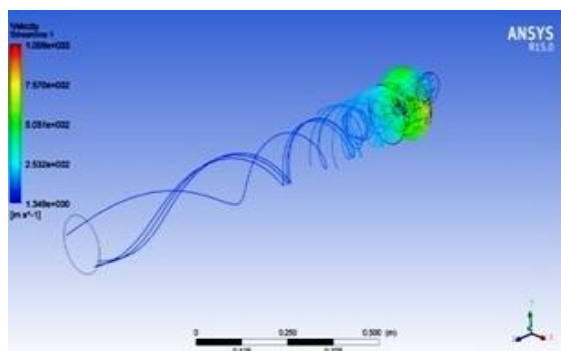
2.6 Back pressure effect

According to [29], powered recovered at low engine speeds is very low for a particular turbine size. Significant energy recovery occurs at 4000-6000 rpm engine operating range. Extreme speed level is associated with high turbine power output and the Mach Number is often supersonic, which invariably does not

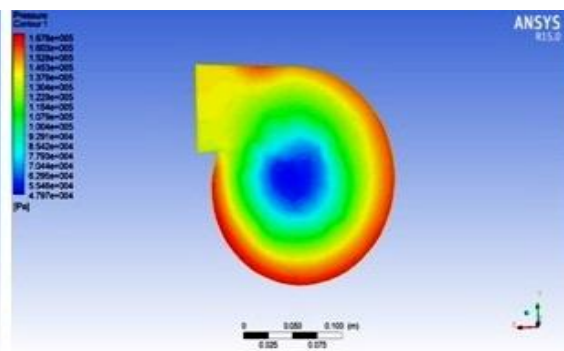
favour the scavenging ability of the engine due largely to associated high back pressure, as much as 30 bars. Since same turbine-engine set-up has been employed for this study, the back pressure effect is minimal since the engine rated speed is not in any way attained.

3.0 RESULTS AND DISCUSSION

The results obtained from the simulation are presented in Figures 4-10.



(a) Velocity streamline plot

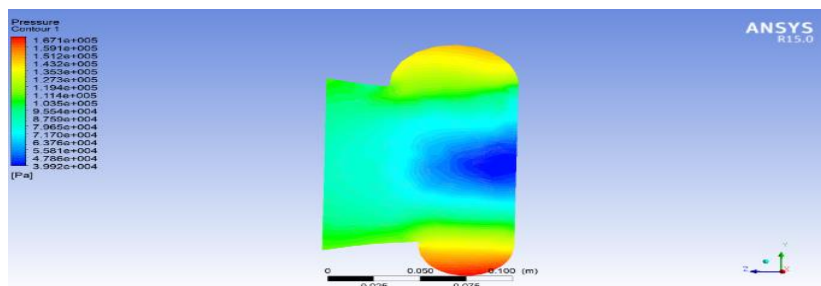


(b) Pressure contour plot on the volume plane XY

Fig. 4: Velocity streamlines and Pressure contour plots on the volute plane XY

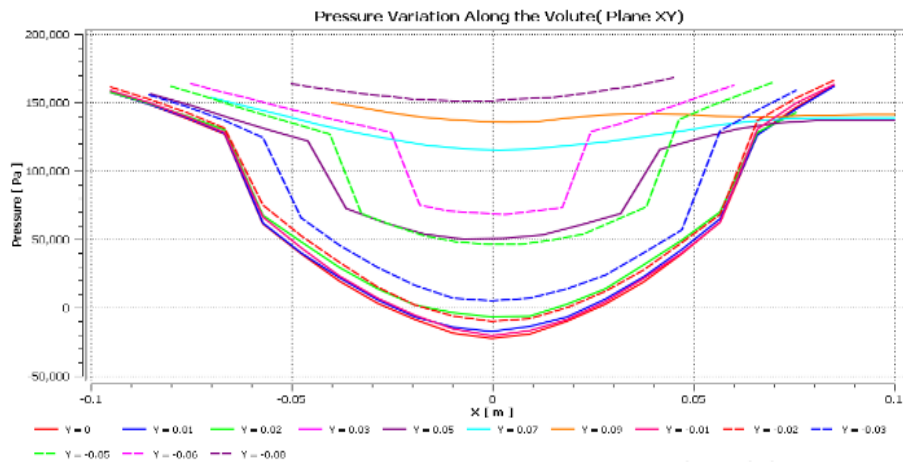
3.1 The volute region

Figures 5(a)-10(b) give indications of the exhaust gas flow property variation throughout the volute region.

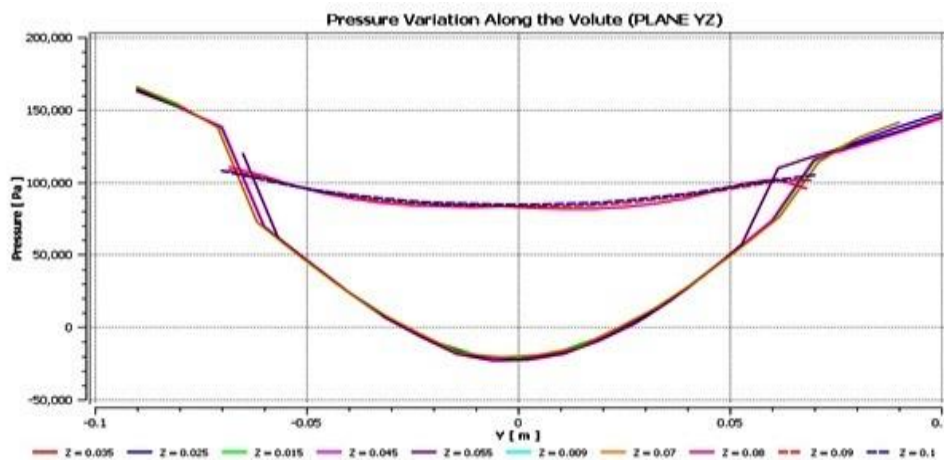


(a) Pressure Contour Plot on Volute Plane YZ

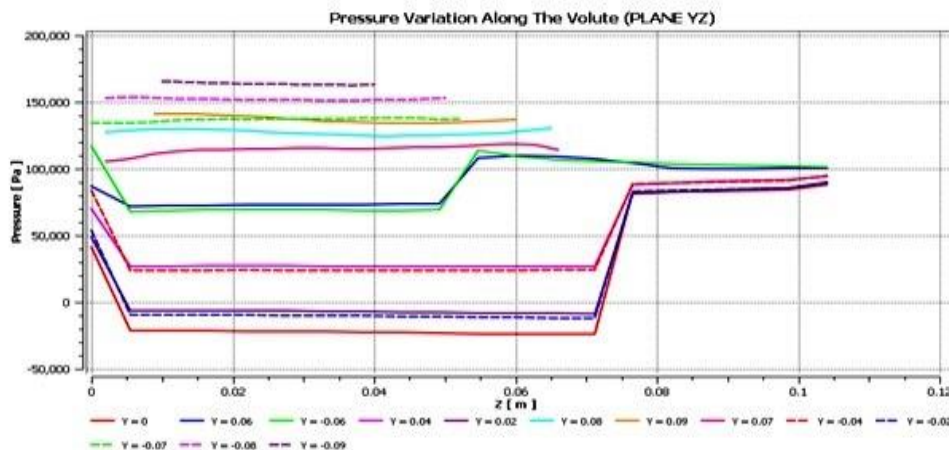
Nwaji et al.,



(b) Pressure Variation in Volute region along the X-axis



(c) Pressure Variation in Volute region along the Y-axis



(d) Pressure Variation in Volute region along the Z-axis

Figs 5(a)-(d) Pressure Contour Plot and variation across different planes in the volute

Figures 4(b)-5(d) show pressure variation contour plots and pressure variations along the coordinate axes in the volute region. The exhaust gas pressure in the volute varied significantly with the x-axis, having its highest values near the walls of the volute and reduced significantly towards the centre of the volute.

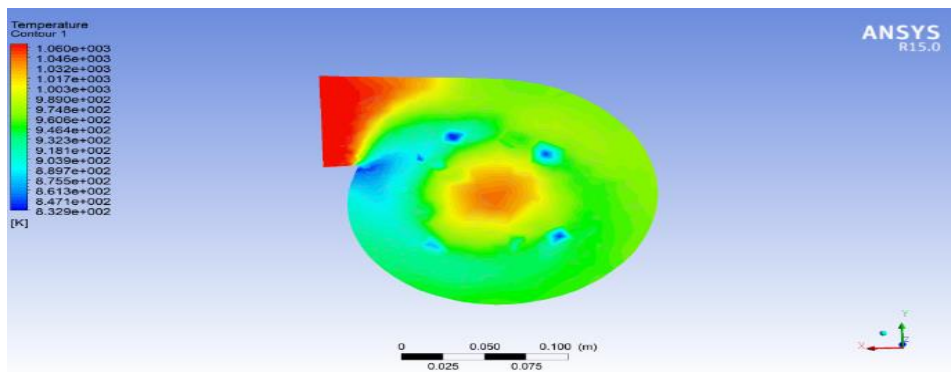
This occurred because the rotor fluid zone is located at the centre of the volute, and since the exhaust gas expanded, its pressure dropped in that zone giving rise to a lower pressure value at the centre of the volute. Supporting this observation is the fact that the average inlet pressure is about 138,447Pa while that of the

Nwaji et al.,

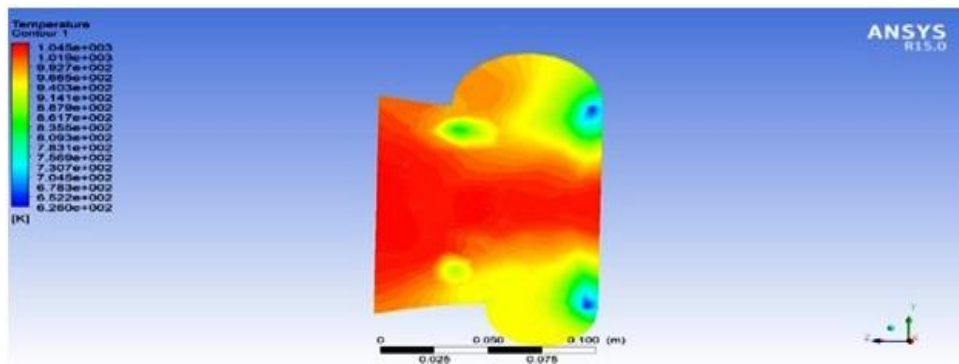
rotor outlet is 19,157.7Pa. The pressure increased significantly as the exhaust gas flowed through the diffuser to an average pressure of 101,640Pa, enabling the fluid to flow out of the volute region. Since a positive pressure differential is required for an outflow, a negative pressure value was noticed very close to the centre of the volute region. This is due to the suction effect created by the shaft hole in the rotor geometry.

From Figures 6(a)-(e), the temperature variation of the exhaust gas in the volute region can be predicted. It is evident that the temperature decreased significantly as the

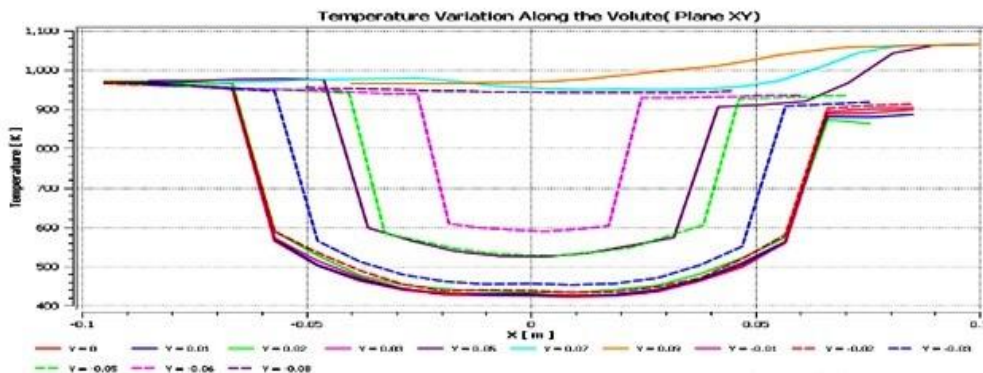
exhaust gas flowed through the volute cross section from a value of about 1064K to 800K before the fluid entered the turbine region. This occurred as a consequence of velocity increment in the volute cross section. Though the contour plots slightly contradicted the graphical plots, it should be noted that the contour plots do not give an actual representation of the flow property variations, but rather an overview of the domain behaviour. Actual representation of flow property variations is given by the graphs as different points on straight lines are gotten and their respective variables plotted against the relevant axes.



(a) Temperature contour plot on volute plane XY

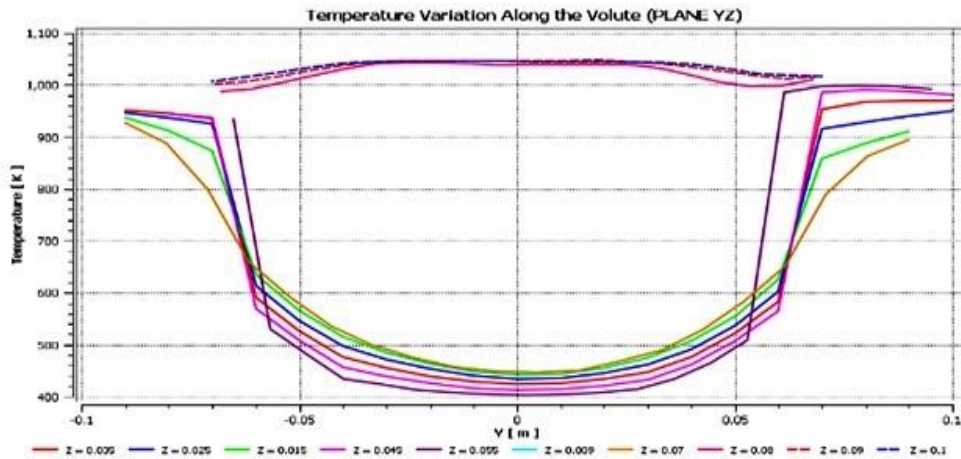


(b) Temperature contour plot on volute plane YZ

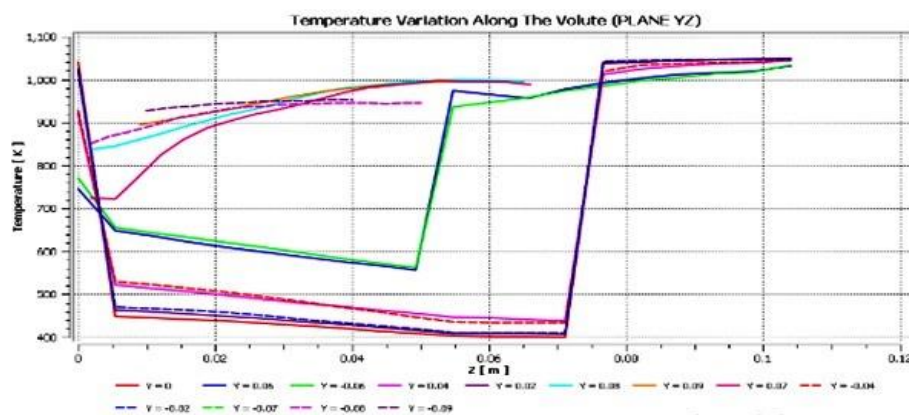


(c) Temperature variation in volute region along the X-axis

Nwaji et al.,



(d) Temperature variation in volute region along the Y-axis

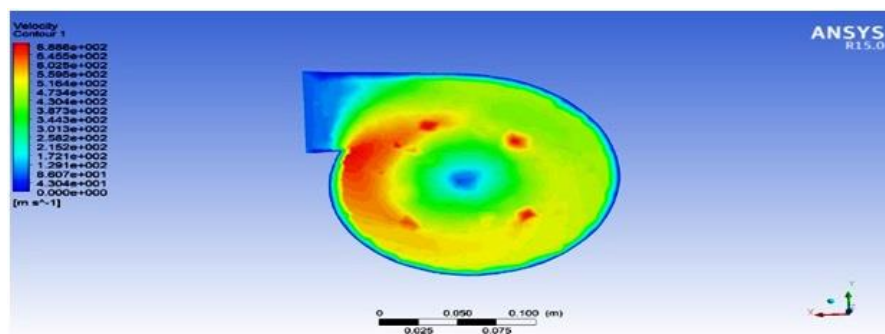


(e) Temperature variation in volute region along the Z-axis

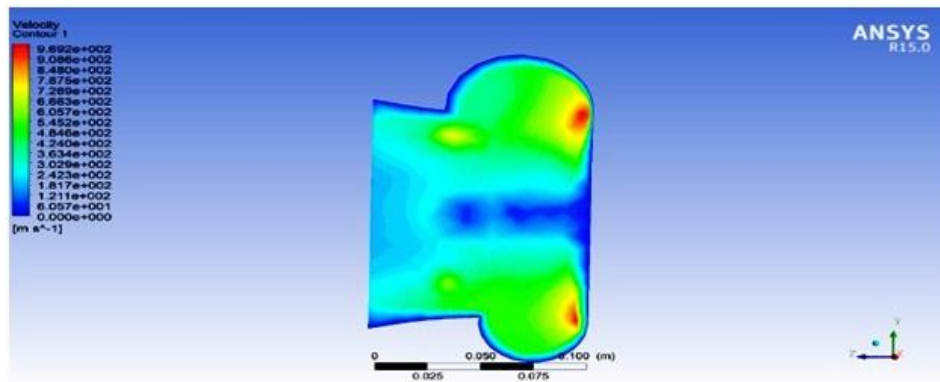
Figs 6(a)-(e) Temperature contour plot and variations along the different planes and axes in the volute region

So, from the temperature graphs in Figures 6(c)-(e), the temperature of the exhaust gas dropped significantly as you move towards the centre of the volute. This is because, the centre housed the portion of the exhaust gas that is rapidly expanding in the turbine region, and as such a reduction in temperature was observed. This is also supported by the fact that the

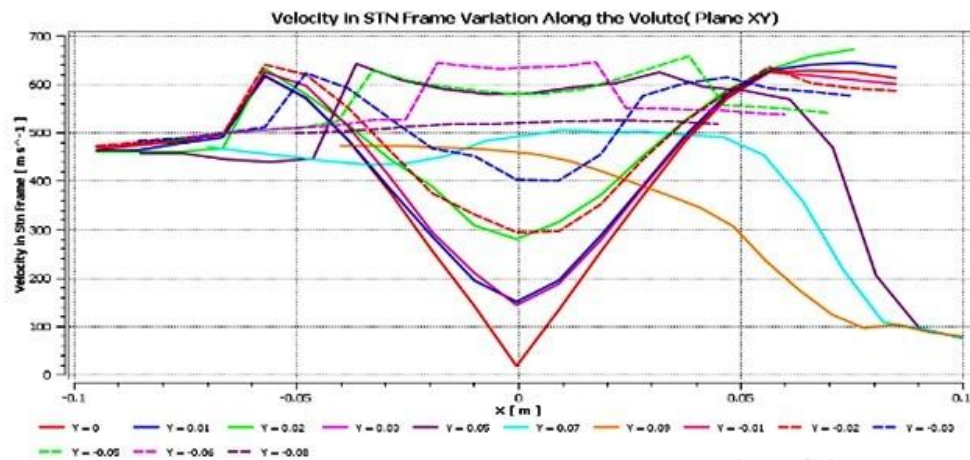
average temperature value at the turbine rotor outlet is 438.49K. The temperature of the exhaust gas also increased as it flowed through the diffuser region due to subsequent increase in pressure. The exhaust gas average diffuser outlet temperature became almost the same as that at the volute inlet.



(a) Velocity contour plot on volute plane XY



(b) Velocity contour plot on volute plane YZ

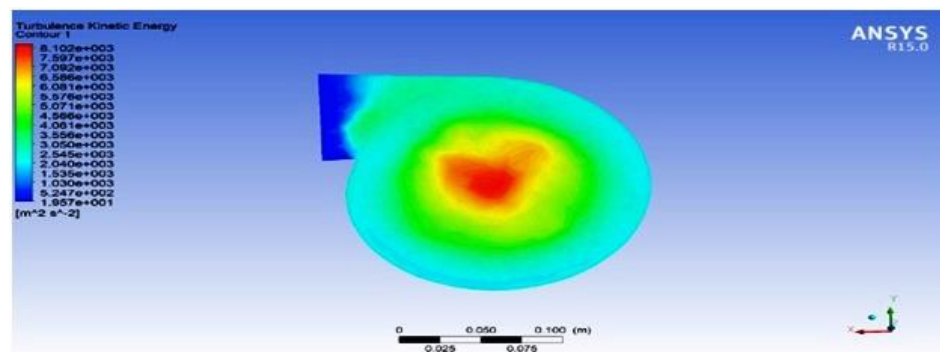


(c) Velocity variation in volute region along the X-axis

Figs 7(a)-(c) Velocity contour plots on different volute planes and velocity variation in the volute region along the X-axis

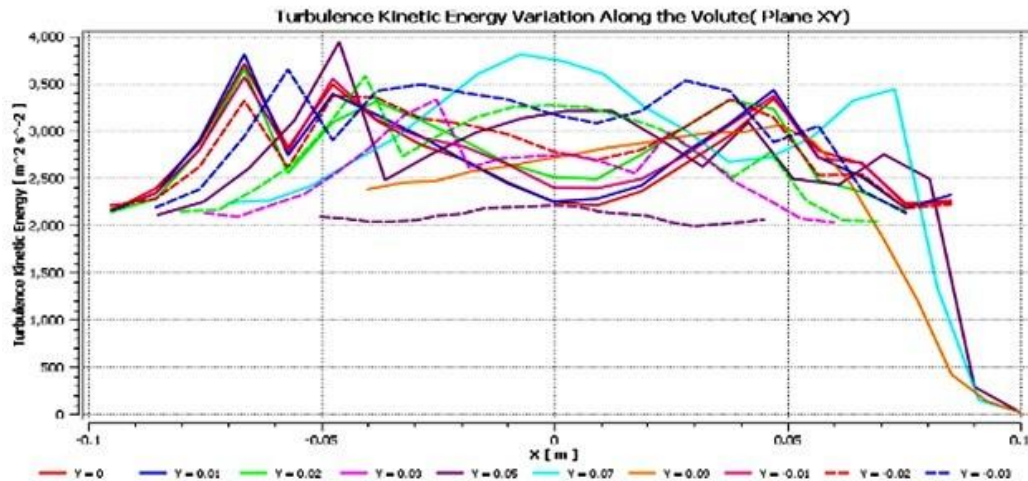
Figures 7(a)-(c) give indications of the exhaust gas velocity variation in the volute region. It can be observed that the velocity was accelerated from about 76.59m/s to 600m/s as the exhaust gas flowed through the cross section at the volute. This arose because of the converging nature of the volute. Although the velocity at the wall of the volute was 0m/s, due to the no slip

condition imposed on the wall, the velocity of the exhaust gas dropped significantly towards the centre of the volute region housing the exhaust gas expanding at the turbine rotor. The velocity of the exhaust gas was also observed to drop as it flowed through the diffuser region, thus supporting the diffuser theory.



(a) Turbulence kinetic energy contour plot on volute plane XY

Nwaji et al.,

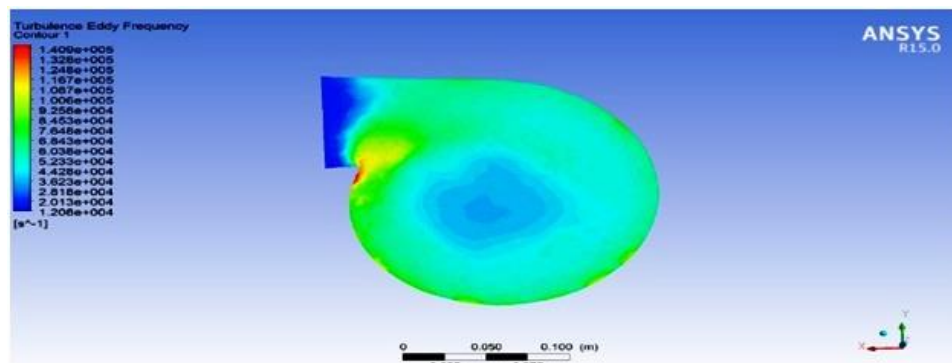


(b) Turbulence kinetic energy variation in volute region along the X-axis

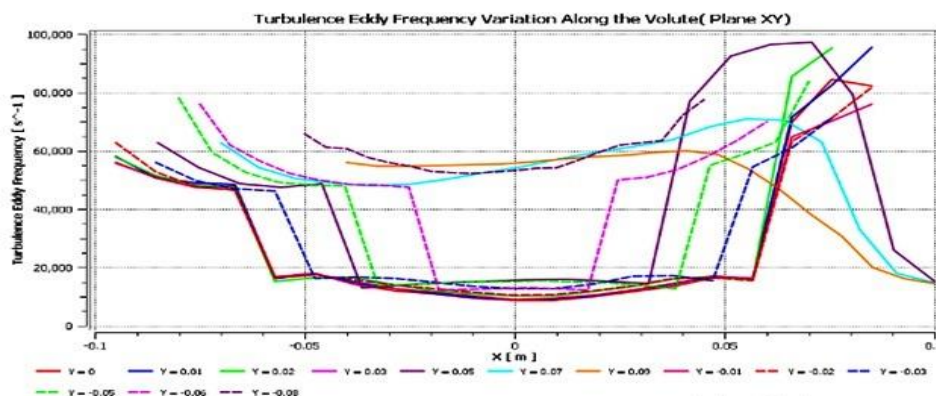
Figs 8 (a) and (b): Contour plot of turbulent kinetic energy and its variation along the X-axis

Figures 8(a) and (b) give the variation of the exhaust turbulence kinetic energy in the volute region. The contour plots and graphs are seen to vary slightly as had been earlier observed. From the graphs, it can be observed that the turbulence kinetic energy exhibits a random variation in the volute region, with maximum and minimum values at different points. This is an expected trend as turbulence entails flow

variable fluctuations. Higher values are observed to occur in the volute flow region while lower values are predominant close to the centre of the volute, showing decreasing turbulence from volute to turbine regions. Therefore, it is established that fluids expanding in a reaction turbine lose part of their energy as they rotate with the turbine rotor.



(a) Turbulence eddy frequency contour plot on volute plane XY



(b) Turbulence eddy frequency variation in volute region along the X-axis

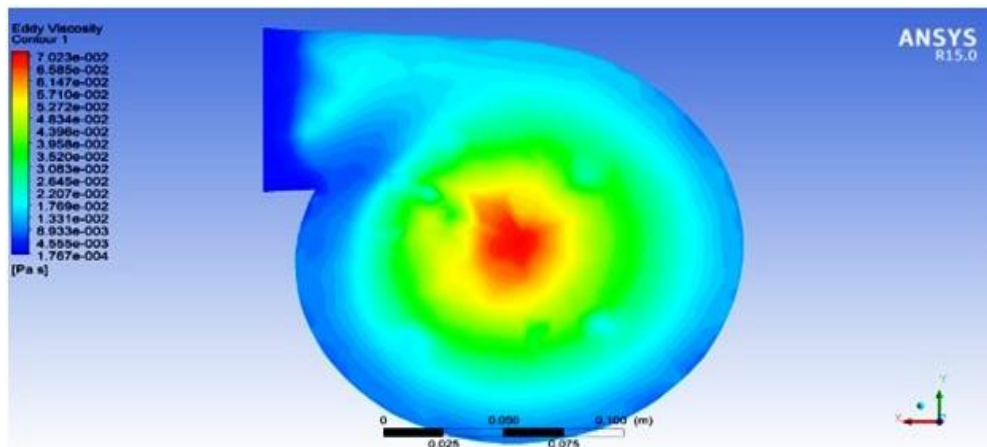
Figs 9(a)-(b): Turbulent Eddy contour plot and variation along the X-axis

Nwaji et al.,

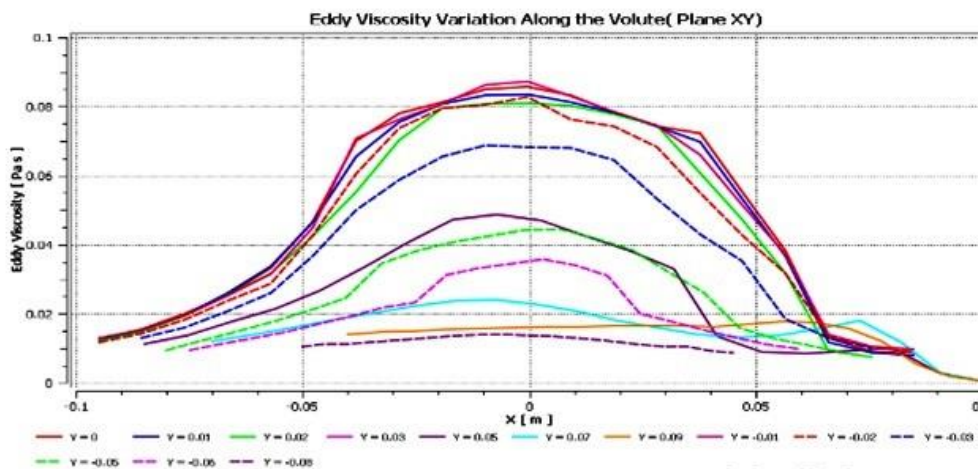
Figures 9(a) and (b) indicate the turbulent eddy frequency variation in the volute region. The turbulent eddy frequency is considered as the rotation time for a turbulent eddy. It was observed to be higher at the volute flow domain and reduced significantly towards the centre of the volute.

Figures 10(a) and (b) give the eddy viscosity variation in the volute region. Eddy viscosity is

considered to be the internal friction between the eddies in a turbulent flow. It gives an indication of the level of turbulence in a flow, and varies inversely with the turbulence intensity. From Figures 9(a) and (b), it is obvious that the eddy viscosity is higher at the centre of the volute than the volute flow region.



(a) Eddy viscosity contour plot on the volute plane XY



(b) Eddy viscosity variation in the volute region along the X-axis

Figs 10(a)-(b): Eddy viscosity contour and variation along the X-axis

CONCLUSION

A computational study on the use of radial turbine for harvesting and converting the hitherto waste energy in the exhaust flue gas of an IC engine was conducted using ANSYS CFX. From the simulation results, it can be seen that a radial flow turbine installed in an IC engine's exhaust gas manifold can be used for the extraction and conversion of the exhaust gas energy into useful work, for the generation of electricity, as about 70% of the exhaust gas energy was extracted with an isentropic

efficiency of 84%. This is evident in the amount of total enthalpy lost as a result of expansion in the turbine. The rotational speed and shaft power of the turbine are fairly high and if coupled to a generator could produce an appreciable electrical power, which can be harnessed to drive vehicle auxiliaries such as the air-conditioner compressor, cooling fans, alternator, etc., thus removing their parasitic effect on the engine and improving engine power. It can also be harnessed in form of turbo-compounding, that is, sending the shaft

power directly to the driveline or incorporated into hybrid cars to charge the batteries when the engine is running. A good beginning would be to build a stationary test unit, having the engine, turbine, cooling chamber or a generator set alternator to study the system's adaptability for cooling or electric power generation, in which case the electric power can be distributed to households or sent to the national grid. The major challenge foreseen is the design of an electric generator that would have the required size and characteristics needed to harness this power. However, this can be overcome by designing appropriate speed reduction mechanisms to match the turbine shaft speed and size with that of the device or intended application. As the ones used in this research were adopted from literature; and since the actual size of turbine and volute geometry used in those literatures could not be determined, a good approximation was used for the research, which should have a minor impact on the accuracy of the results obtained.

Acknowledgements

We wish to appreciate the Mechanical Engineering Department, Federal University of Technology, Owerri, Nigeria and the African Centre of Excellence for Sustainable Power and Energy Development (ACE-SPED), University of Nigeria Nsukka, for her support during this research. Moreover, we also acknowledge all the authors cited in this work as well as the reviewers for their valuable comments.

REFERENCES

- [1] Okonkwo, P.A., Omenihu, I, 2021. Production and Characterization of Biodiesel from Mango Seed Oil (MSO), Nigerian Journal of Technology (NIJOTECH), 40(4) 598-607.
- [2] Nwaji, G.N., Udongwo, S., Ajayi, K.T., Nwufo, O.C., Onwuachu, C.C., Ofong, I. 2021. Investigation of the Effect of Engine Speed on the Radial Inflow Turbine for Automotive Exhaust Energy Recovery. International Journal for research in Engineering & Technology 1(1) 1-15.
- [3] Ononogbo, C. Nwufo, O.C., Okoronkwo, C.A., Ogueke, N.V., Igbokwe, J.O., Anyanwu, E.E, 2020. Equipment sizing and method for the application of exhaust gas waste heat to food crops drying using a hot air tray dryer. Indian Journal of Science and Technology, 13, 502-518.
- [4] Lee, M.Y., Seo, J.H., Lee, H.S., Garud, K.S. 2020. Power generation, efficiency and thermal stress of thermoelectric module with leg geometry, material, segmentation and two-stage arrangement. Symmetry, 12, 786.
- [5] Saidur, R, 2012 Technologies to recover exhaust heat from internal combustion engines. Renewable and Sustainable Energy Reviews. 16, 5649-5659.
- [6] Nadaf, S.L., Gangavati, P.B, 2014. A review on waste heat recovery and utilization from diesel engines. International Journal of Advanced Engineering Technology 4, 31-39.
- [7] Oyedepo, S. O., Fakeye, B. A, 2021. Waste heat recovery technologies: pathway to sustainable energy development. Journal of Thermal Engineering, 7 (1) 324-348.
- [8] Ajayi, K.T., Nwaji, G.N., Ojo, B.O, 2014. Numerical study of the properties of high energy gas flow through a microturbine in 3D cascade, Journal of Emerging Trends in Engineering and Applied Sciences, 5(7) 88-94.
- [9] Ajayi, K.T., Nwaji, G.N, 2010. Numerical Investigation of Internal Combustion Engine, International Journal of Engineering, 4(2) 229-239.
- [10] Nwaji, G.N., Udongwo, S., Ajayi, K.T., Uguru-Okorie, D.C., Anyanwu, E.E, 2021 Numerical Simulation of Flow across a Radial Turbine for Prediction of Shaft Power for Driving Automobile Air Conditioners, Int. J. Adv. Sci. Eng. 7(4) 1904-1911.
- [11] Pradip, G. K., Hole, J. A, 2015. A review on waste heat recovery and utilization from exhaust gas of I.C engine. International Journal for Scientific Research & Development, 3, 1321-1325.
- [12] Hatazawa, M., Sugita, H., Ogawa, T., Seo, Y., 2004. Performance of a thermo acoustic sound wave generator driven with waste heat of automobile gasoline engine. Transactions of the Japan Society of Mechanical Engineers, 70, 292-299.
- [13] Stabler, F, 2002. Automotive Applications of high efficiency thermoelectric. Proceedings of the DARPA/ONR program

- review and DOE high efficiency thermoelectric workshop, San Diego. 1–26.
- [14] Woolley, E., Yang, L., Alessandro, S. 2018. Industrial waste heat recovery: A systematic approach. *Sustainable Energy Technologies and Assessments*, 29, 50–59.
- [15] Ononogbo, C., Nwufu, O.C., Nwaji, G.N., Nwosu, E.C., Nwadinobi, P.C., Okoronkwo, C.A., Igbokwe, J.O., Anyanwu, E.E, 2022, Investigation of the Thermal Profile of a Crop Dryer Powered by Generator Exhaust Gas Waste Heat. *Int. J. Adv. Sci. Eng.*, 8 (3) 2235-2241.
- [16] Reshid, M. N., Muhamad, W.M.W., Majid, M.A.A, 2017. Transient Simulation of a Waste Heat Recovery from Gas Turbine Exhaust. *Journal of Applied Sciences*, 17 (1): 22-31.
- [17] U.S. Department of Energy, 2008. Waste Heat Recovery: Technology and Opportunities in U.S. Industry, Office of Energy Efficiency and Renewable Energy, Industrial Technologies Program (ITP).
- [18] Bergmeier, M. 2003. The history of waste energy recovery in Germany since 1920. *Energy*. 13, 1359–1374.
- [19] Reddy, C., Naidu, S, 2013. Waste Heat Recovery Methods and Technologies, University of Singapore, 1 1 2013. <http://www.chemengonline.com/waste-heat-recovery-methods-and-technologies/?printmode=1>.
- [20] Naik-Dhungel, N, 2012. Waste Heat to Power Systems. https://www.epa.gov/sites/default/files/201507/documents/waste_heat_to_power_systems.pdf
- [21] Jouhara, H., Navid, K., Sulaiman, A., Bertrand D., Amisha C., Savvas, A. T. 2018. Waste heat recovery technologies and applications. *Thermal Science and Engineering Progress*, 6, 268–289.
- [22] Luddecke, B., Filsinger, D., Ehrhard, J. 2012. On Mixed Flow Turbines for Automotive Turbocharger Applications, IHI charging Systems International GmbH, Engineering Division, Haberstrabe 24, D-69126 Heidelberg, Germany.
- [23] Yang, M., Martinez-Bofas, R., Rajoo, S., Yokoyama, T., Ibaraki, S. 2015. An Investigation of Volute Cross-Sectional Shape on Turbocharger Turbine Under Pulsating Conditions in Internal Combustion Engine, *Energy Conversion and Management* 105, 167-177.
- [24] Shah, S.P., Channiwal, S.A., Kulshreshtha, D.B., Chaudhari, G, 2014. Design and Numerical Simulation of Radial Inflow Turbine Volute, *International Journal of Turbo & Jet-Engines*, 31(4) 287-301.
- [25] Cristian, C. 2012. Three Dimensional CFD Simulation of a Turbocharger Turbine for Motorsport Applications, M.Sc. Thesis, Università Degli Studi Di Parma, Stuttgart.
- [26] Galindo, J., Fajardo, P., Navarro, R., Garcia-Cuevas, L.M. 2013a. Characterization of a Radial Turbocharger in pulsating flow by means of CFD and its application to Engine Modelling. *Applied Energy* 103, 116-127
- [27] Galindo, J., Hoyas, S., Fajardo, P., Navarro, R. 2013b. Set-up Analysis and Optimization of CFD Simulation for radial flow turbine. *Engineering Applications of Computational Fluid Mechanics*, 7(4) 441-460.
- [28] ANSYS Inc. (2013). ANSYS CFX-Solver Theory Guide: Modelling Turbulent Flows. Release 15.0.
- [29] Nwaji, G.N., Udongwo, S., Ajayi, K.T., Uguru-Okorie, D.C., Anyanwu, E.E. 2021. Numerical Simulation of Flow across a Radial Turbine for Prediction of Shaft Power for Driving Automobile Air Conditioners, *Int. J. Adv. Sci. Eng.* 7(4) 1904-1911.